International Journal of Engineering Inventions
e-ISSN: 2278-7461, p-ISSN: 2319-6491
Volume 14, Issue 7 [July 2025] PP: 31-36

Variation of cold fluid flow rate in a Shell and Tube type Heat Exchanger with square geometry using the Computational Fluid Dynamics method

Ida Bagus Alit*, IGNK Yudhyadi
Department of Mechanical Engineering, Faculty of Engineering, University of Mataram, Jl. Majapahit No. 62 Mataram-Nusa Tenggara Barat 83125, Indonesia.
*Corresponding Author: **alit.ib@unram.ac.id***

ABSTRACT: *A heat exchanger has various design models, one example being the shell and tube type heat exchanger. This device is useful for increasing or decreasing temperature. One of its applications in the hydroelectric power industry is as an oil cooler bearing. This study aims to design and simulate a shell and tube type heat exchanger with tube inner diameters of 20 mm and 25 mm, arranged in a square configuration. The design results show a total length of 1318.59 mm, an outer shell diameter of 300 mm, and an inner shell diameter of 320 mm. Simulations were carried out with variations in cold fluid flow rates of 0.3, 0.4, 0.5, and 0.6 kg/s. The research results show that the highest heat transfer coefficient occurs at a cold fluid inlet mass flow rate of 0.6 kg/s, corresponding to a value of 98.96 W/m²K. As the inlet fluid flow rate increases, the heat transfer coefficient also increases. The highest efficiency is achieved at a cold fluid mass flow rate of 0.6 kg/s, reaching 70.04%.*

Date of Submission: 02-07-2025 Date of acceptance: 12-07-2025

I. INTRODUCTION

Heat exchangers are in high demand across industrial companies, factories, and power plants due to the large production volumes required in these sectors. This demand continues to grow alongside the rapid expansion of various industries, which also leads to an increased need for substantial energy supplies. One major source of electrical energy is hydroelectric power plants, which utilize the flow of water to generate electricity. These plants function by converting the kinetic energy of moving water (sourced from dams or waterfalls) into mechanical energy through water turbines, which is then transformed into electrical energy using generators. The process of heat transfer between two fluids at different temperatures, separated by a solid barrier, is a common phenomenon in numerous engineering applications [1], [2]. In power plants, one engineering application that relies on heat exchangers is the cooling of turbine bearings, which are essential for maintaining the alignment of the turbine shaft along a single axis. To ensure the proper functioning of these bearings, a cooling system that uses oil lubrication is implemented [3]. The cooling of this lubricating oil is typically achieved using a shell and tube heat exchanger [4]. This type of heat exchanger consists of a shell, which serves as the outer casing, and tubes housed within the shell. Heat transfer occurs as fluids at different temperatures flow through these separate sections [5]. The flow of these fluids can be arranged in various configurations, including parallel flow, counterflow, crossflow, or a combination of these patterns [6]. In parallel flow, both fluids enter from the same side, move in the same direction, and exit together. In contrast, counterflow involves the fluids entering from opposite ends, flowing in opposite directions, and exiting through separate outlets [7]. In a shell-and-tube heat exchanger, one fluid flows through the tubes while another fluid circulates around the outside of the tubes within a cylindrical enclosure known as the shell. The tubes are positioned parallel to the shell's axis to optimize the flow arrangement [8]. To enhance the velocity and efficiency of fluid flow on the shell side, baffles are installed. These components also help improve the accuracy of predictions related to pressure drop and heat transfer within the tube section [9]. Pressure drop, or pressure loss, occurs due to friction between the flowing fluid and the inner walls of the tubes.

Heat transfer in heat exchangers can be analyzed using both experimental methods and simulations. Previous studies have investigated flow patterns in short heat exchangers with and without baffles [10], as well as the impact of varying the number of tubes and baffles on the performance of shell-and-tube heat exchangers [11]. In these studies, researchers explored three different configurations of tube numbers, baffle arrangements, and fluid flow rates. The findings revealed that changes in the number of tubes and baffles significantly influence the overall heat transfer coefficient and the efficiency of the heat exchanger. However, a notable drawback of experimental research is that it can be time-consuming and costly.

Computational Fluid Dynamics (CFD) is a widely used method for analyzing flow simulations. Numerical simulations based on computational methods, supported by CFD, are employed primarily because they allow for the evaluation of testing parameters without the need for physical experiments [12]. This approach helps in analyzing temperature distribution patterns within shell-and-tube heat exchangers and assessing their effects on various tube configurations, such as triangular, rotated triangular, square, and rotated square arrangements [13].

Given this background, it is essential to conduct simulations of shell-and-tube heat exchangers using SolidWorks. The design of the heat exchanger, including tube diameters and arrangements, influences heat transfer, resulting in performance variations compared to other heat exchanger types. In this study, heat transfer simulations were performed using SolidWorks to evaluate the effectiveness of shell-and-tube heat exchangers with different tube diameters and configurations. SolidWorks provides simulation results that closely match actual experimental data. Additionally, simulated testing significantly reduces the time and costs associated with physical experiments, enhancing overall efficiency. The material properties used in this research were sourced from the material library available within the SolidWorks application.

II. EXPERIMENTAL SETUP

The geometry used is standard from the Tubular Exchanger Manufacturers Association (TEMA) standard [14]. The tube arrangement used is square with a total of 6 tubes. The following are the geometric variations of the tube arrangements designed using SolidWorks in this study:

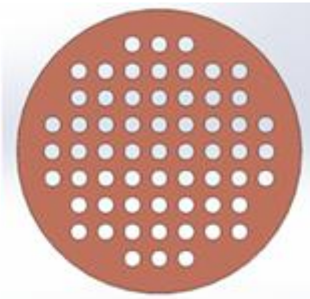


Figure 1. Tube arrangement

In the heat exchanger planning process, a heat load is required to be charged to the heat exchanger. The following operational data will be used as a heat exchanger load.

Table 1. Operational data of heat exchangers

Cooled fluid	Lubricating oil
inlet temperature	59^0 C
outlet temperature	35^0 C – 45^0C
Flow rate	0.2 kg/s
Cooled fluid	water
inlet temperature	26^0 C
Flow rate	0.3 ; 0.4 ; 0.5 and 0.6 kg/s

Meshing involves breaking down objects into smaller elements for simulation modeling. This process is carried out based on the geometry of the control volume, with a hexahedral mesh type being used. The analysis focuses on the flow characteristics of fluids both inside the shell and around the tubes. The modeling process proceeds by generating a mesh, which divides the model into smaller segments for detailed analysis. Prior to selecting the mesh size, a convergence test is performed to identify an optimal mesh size that ensures stable and accurate simulation results.

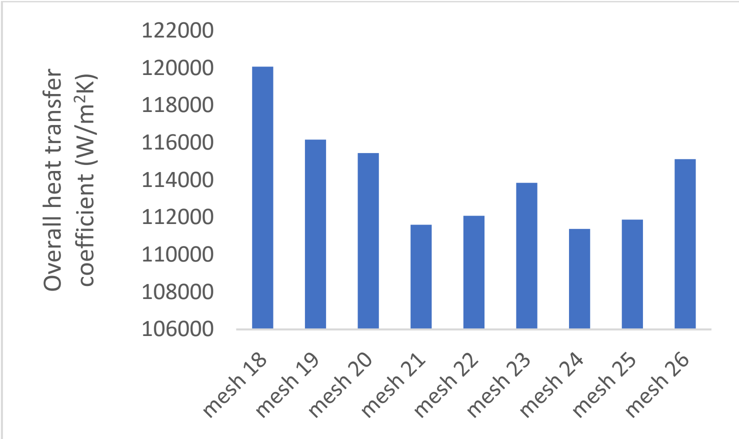
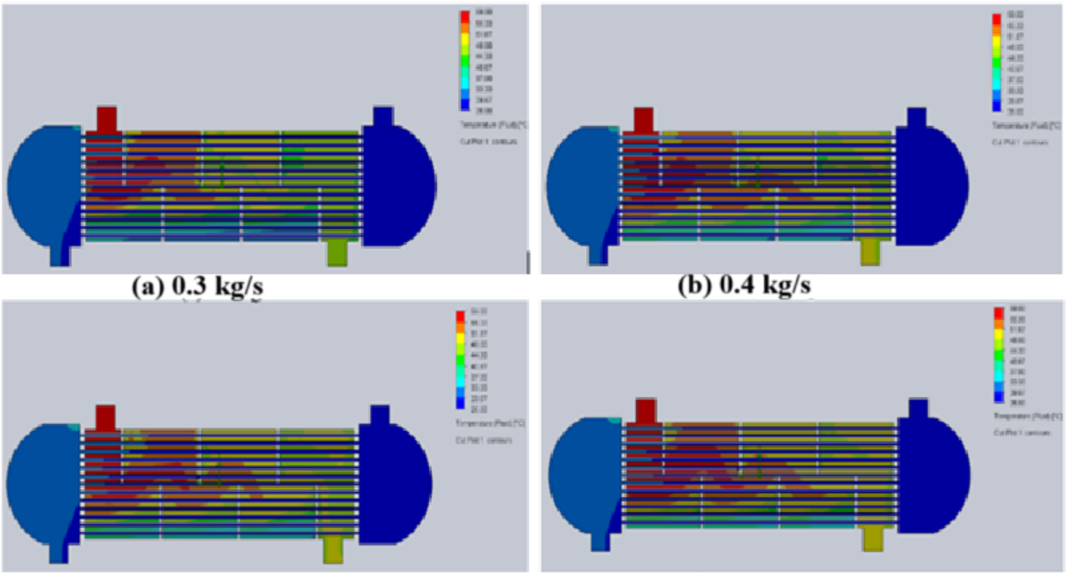


Figure 2. Mesh Convergence Test Results

III. RESULTS AND DISCUSSION

The image below shows a visualization obtained from the simulation results of a temperature distribution cross-section plot, taking into account the tube arrangement, an inlet mass flow rate of 0.2 kg/s for the hot fluid, and variations in the flow rate for the cold fluid.



(a) 0.3 kg/s **(b) 0.4 kg/s**

(c) 0.5 kg/s **(d) 0.6 kg/s**

Figure 3. Cut plot of temperature distribution with variations in the cold fluid flow rate

The deep blue color represents the lowest bottom temperature. The cold fluid enters the tube at 26°C and, as it flows through, experiences friction with the pipe's inner surface, initiating heat transfer. Meanwhile, the hot fluid, at 59°C, moves along the outside of the pipe, transferring heat from the tube's outer wall. This heat is then conducted to the cold fluid inside the tube.

Near the inlet, the hot fluid maintains a higher temperature compared to the fluid closer to the shell outlet. As a result, the absorbed heat is transferred through convection, moving in a one-dimensional manner toward the cold fluid.

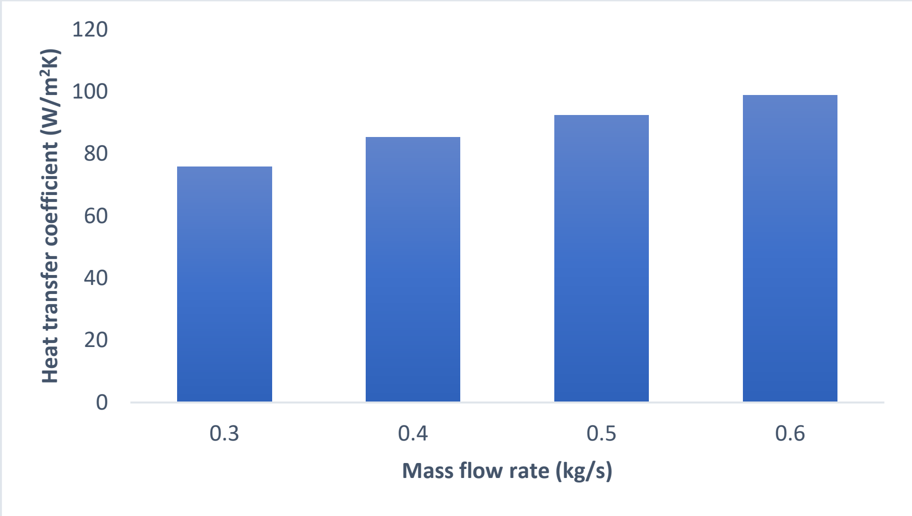


Figure 4. Graph of the overall heat transfer coefficient

Figure 4 shows a graph of the overall heat transfer coefficient for variations in the inlet mass flow rate for the hot fluid. Based on the simulation results, the highest heat transfer coefficient occurs at a cold fluid inlet mass flow rate of 0.6 kg/s, reaching 98.96 W/m²K. When the inlet fluid flow rate increases, the heat transfer coefficient also increases. This is due to the fact that higher flow velocities lead to increased flow turbulence. The overall heat transfer coefficient is influenced by the high heat transfer that occurs in hot and cold fluids, fluid flow rate, tube cross-sectional area, and logarithmic average temperature difference (ΔTlm).

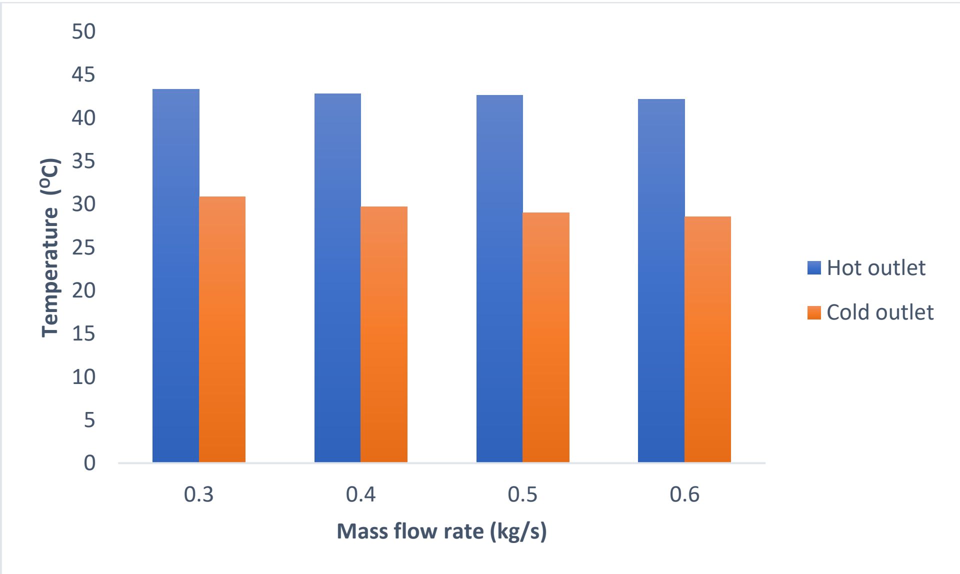


Figure 5. Graph of hot and cold fluid outlet temperature distribution

The simulation results show that the higher the mass flow rate of the incoming cold fluid, the higher its outlet temperature. This occurs because as the fluid flow rate increases, a larger mass of fluid receives heat transfer, resulting in a lower temperature rise. The lowest temperature increase was observed at a cold fluid mass flow rate of 0.3 kg/s, ranging from 26°C to 28.57°C (9.9%), while the highest increase occurred at a mass flow rate of 0.6 kg/s, reaching 18.6%.

The outlet temperature of the hot fluid decreases as the cold fluid flow rate increases. This is due to the higher heat transfer coefficient, which causes more heat to be transferred from the hot fluid to the cold fluid. This observation is supported by the visual representation in Figure 3, which shows that an increase in the hot fluid mass flow rate results in greater and longer heat propagation in the heat exchanger. The increase in turbulence at higher fluid mass flow rates contributes to this phenomenon, leading to more efficient heat transfer.

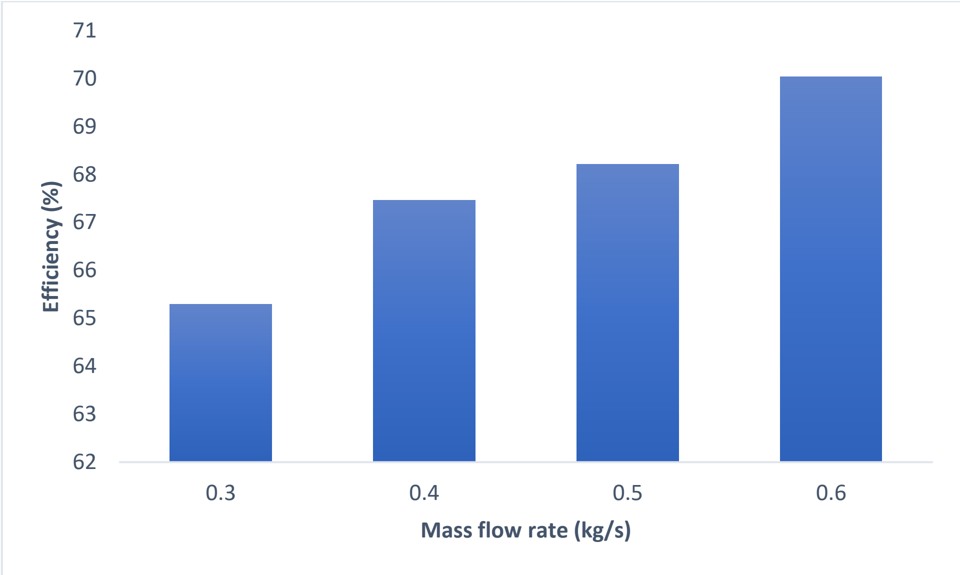


Figure 6. Graph of shell and tube heat exchanger effectiveness

The image above shows the effectiveness of the heat exchanger against variations in the mass flow rate of the cold fluid. The simulation was carried out with a constant hot fluid flow rate of 0.2 kg/s. The research results showed that the highest efficiency was achieved at a cold fluid mass flow rate of 0.6 kg/s reaching 70.04%, while the lowest efficiency was 65.29% at a cold fluid mass flow rate of 0.3 kg/s. The efficiency of a shell and tube heat exchanger is directly proportional to the mass flow rate of the incoming cold fluid.

IV. CONCLUSION

The velocity of hot and cold fluid flow affects the overall heat transfer coefficient. Simulation results for variations in cold fluid inlet flow rates show that higher cold fluid inlet velocity increases the heat transfer coefficient. The highest heat transfer coefficient occurs at a cold fluid inlet mass flow rate of 0.6 kg/s, reaching 98.96 W/m²K. The outlet temperature of the hot fluid decreases as the cold fluid flow rate increases. This occurs due to an increase in the heat transfer coefficient, allowing more heat to be transferred from the hot fluid to the cold fluid. The maximum efficiency of 70.04% is achieved when the cold fluid mass flow rate reaches 0.6 kg/s, while the minimum efficiency of 65.29% occurs at a cold fluid mass flow rate of 0.3 kg/s. The efficiency of a shell-and-tube heat exchanger increases with a higher cold fluid mass flow rate.

REFRENCES

[1] Selbaş, R., Kızılkan, Ö., & Reppich, M. (2006). A new design approach for shell-and-tube heat exchangers using genetic algorithms from economic point of view. Chemical Engineering and Processing: Process Intensification, 45(4), 268-275.

[2] De, D., Pal, T. K., & Bandyopadhyay, S. (2017). Helical baffle design in shell and tube type heat exchanger with CFD analysis. International journal of heat and technology, 35(2), 378-383.

[3] Sembiring, Adelia F. Br., Siahaan, Johan Cristian., Siahaan, Sihar. (2022). Analysis of the horizontal francis turbine bearing cooling system unit 1 with 6.7 MW power and 750 rpm rotation at PLTA Pakkat PT. Synergy Sakti Sentosa. Journal ilmiah teknik mesin 03 (01)

[4] Incropera, Frank P., Dewitt, David P., Bregman, Theodore L., Lavine, Adrienne S. (2007). Fundamentals of heat and mass transfer 7thedition. United State of America. John Wiley & sons, Inc.

[5] Arani, A. A. A., & Moradi, R. (2019). Shell and tube heat exchanger optimization using new baffle and tube configuration. Applied Thermal Engineering, 157, 113736.

[6] Prithiviraj, M., & Andrews, M. J. (1998). Three dimensional numerical simulation of shell-and-tube heat exchangers. Part I: foundation and fluid mechanics. Numerical Heat Transfer, Part A Applications, 33(8), 799-816.

[7] Adhitiya, Anggareza dan Ichsani, Djatmiko. (2013). Simulation of the performance of a shell and tube type heat exchanger with double segmental baffles against helical baffles. Journal Teknik POMITS, 2 (3) ISSN: 2337-3539

[8] Prasad, A. K., & Anand, K. (2020). Design Analysis of Shell Tube Type Heat Exchanger. Int. J. Eng. Res. Technol, 9(01).

[9] Gustyawan, Bayu Esha. (2017). Helical baffle variation analysis on shell and tube heat exchanger based on Computational Fluid Dynamics (CFD) [Final Project. Sepuluh Nopember Institute of Technology (ITS) Surabaya].

[10] Pal, E., Kumar, I., Joshi, J. B., & Maheshwari, N. K. (2016). CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles. Chemical engineering science, 143, 314-340.

[11] Irawan, Dwi., Mafruddin., Rian., Wibowo, Mukti., Anggara, Zul. (2020). Effect of the number of tubes and baffles on the effectiveness of the shell and tube heat exchanger. SNPPM-2 Muhammadiyah Metro University. ISBN 978-623-90328-5-2

[12] Affandi, Nurlan. (2018). Performance simulation of a shell and tube type heat exchanger with helical baffles and disk and donut baffles. JTM 06 (01), 61-68

[13] Alit, I. B., & Yudhyadi, I. G. N. K. (2023). Simulation of shell and tube type heat exchanger for square geometry using computational fluid dynamics method. *World Journal of Advanced Engineering Technology and Sciences*, *9*(2), 345-350.

[14] Byrne, Richard C. (2019). Standards of Tubular Exchanger Manufacturers Association 10th (TEMA). Tubular Exchanger Manufacturers Association, Inc.